

Project Name: Structural Strength Evaluation for Basemachine of a Boom Truck

Software: Hyperworks, Ansys APDL

Scope: To evaluate the structural strength of a T110 Basemachine with different boom orientations.

- Creation of Finite Element Model with 2D Shell & 3D solid hexahedral elements at subsystem level for Superstructure, Chassis with Hub & Outriggers.
- Weld bead modeling in solid hexahedral elements to identify the stresses at weld locations.
- Extension cylinders and bolts were modeled through 1D elements assigning their respective cross sections.
- Identified critical loading configurations from loadcharts.
- FEA was carried out using advanced frictional contact simulations.
- Validated pin and cylinder forces through hand calculations.
- Design recommendations are shared with thorough validation on feasibility and costs associated with manufacturing
- Optimized the structures using FEA in par within the target weight and costs.
- Buckling Analysis is carried out to evaluate the structure for Buckling Strength.







Project Name: Finite Element analysis of Pick & Carry crane to optimize

Software: Hypermesh, Hyperview, ANSYS Classic Scope: To optimize the structure to reduce the overall machine weight & material cost without compromising on the performance





- Boom and chassis frames concept were studied with inclusion of rope, extension cylinders, luffing cylinder & wear pad flexible contacts
- High stress regions on boom sections , rear chassis & front chassis are analyzed for critical 14 load cases and proper reinforcement has been added to strengthen the structures.
- Extensive study was carried out to finalize the plate thickness and material grade to optimize the structure
- Finite element analysis results were evaluated wrt. hand calculations at all reaction points and cross sections



Baseline Chassis Frame

Final Chassis Frame



Project Name: : Structural analysis of 1700R skid steer

Software: ANSYS Workbench

Scope: Evaluating the chassis of 1700R skid steer for different working conditions of loader arm and bucket using FEA simulation



- External loads considered in the analysis include cabin ROPS weight, engine weight, cooling tank weight, fuel tank, hydraulic tank, gear pump, front drive line, main pump and unaccountable mass.
- Side load and Tractive load wre calculated based on maximum lift load derived from iterative analysis.
- Different load were considered for the analysis to study deformation and stress intensities on the chassis.
- Its observed that stress on the chassis were not so predominant over all loading condition for different positions.



 Maximum lift and tilt cylinder pressures is within the system pressure limit of 190 bar for all load cases



Project Name: : Structural validation of AWP machine

Software: ANSYS Workbench, Analytix

Scope: Static structural analysis of boom lift structure with friction contact methodology and to redesign the structures if the members failing to pass the allowable stress limits



- To perform FEA (Finite Element Analysis) of boom lift structure as per ANSI (52 Load Cases) and EN280 (104 Load Cases) standards
- ANALYTIX, kinematic tool is used to extract the Load data and to evaluate the pin forces of analyzed model
- Frictional contact method is used to transfer the loads between interacting bodies, this method ensures best co-relation with physical testing conditions and load extracting tools



Project Name: : Scissor lift NPD Project

Software: ANSYS Structural

Scope: To support design and development of scissor structures, verities of load cases to be analyzed for concept model and for optimized structure.



- Assembly level analyses were carried out
 for the standards ANSI and EN280
 standards to qualify for the market
 requirements
- Standard procedures and common friction contact method are established to simulate 18 load cases in stipulated time
- All loads were derived from Analytix and the results were evaluated w.r.t. Analytix tool
- Optimization of the cylinder was done by finding ideal cylinder kinematic points using design of experiment
- Complete scissor analysis reference document created for future scissor analysis
- 46ft link set with two cylinder analysis , was most challenging due to common hydraulic pressure supply for both top and bottom cylinders









Project Name: Weld sizing of the joint

Software: ANSYS Workbench and Excel Scope: Weld sizing of the joint in operating and service condition.



- Prior to weld joint analysis in FEA, minimum weld leg length required at joint are calculated considering matching or under matching material depending on the requirement.
- Operating and service condition load cases are considered for calculations.
- Weld thickness calculated from the critical case and fabricating feasibility are used to finalize the weld dimensions in the assembly.





Project Name: Fatigue estimation of AWP machines

Software: ANSYS Workbench,

Scope: Fatigue analysis of boom lift structure & redesign the structures if the welding joints per weldment failing to qualify the allowable fatigue factor





- Perform fatigue analysis of boom lift structure as per BS_EN_13001-1-3_2015 (9 Load Cases) standards
- ANALYTIX, kinematic tool is used to extract the Load data and to evaluate the pin forces of analyzed model
- Critical locations are identified based on the vector plots for all the weld joint in a weldments
- Fatigue calculation parameters are selected based on the type of weld joint identified from the standards
- Based on vector plots, maximum & minimum principle stress ranges considered to calculate the fatigue factor.

